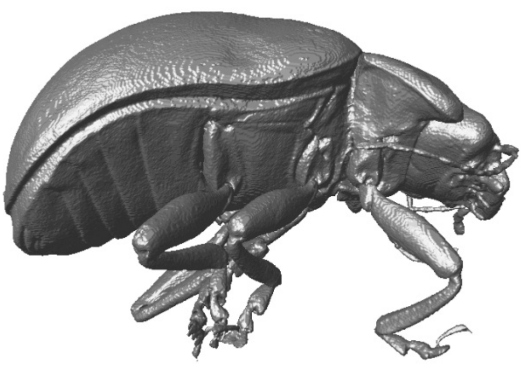
CT-Volume
(version 2.0)

The user's guide

Contents



1. Introduction

CT-volume ("CTVol") is an application for viewing and manipulating 3d surface rendered models from micro-CT scans. The 3d models are created in the program CT-analyser ("CTAn"). The purpose of these models is to make visible and tangible aspects of the 3D structure of an object that has been imaged by micro-CT. Single and multiple models can be viewed. Multiple models can be made from the same scan of one object, for instance to visualise different material components of the object having differing x-ray opacity, using different visual properties such as colour and transparency to make the components simultaneously visible. Visualisation can be enhanced further by the creation of animated movies.

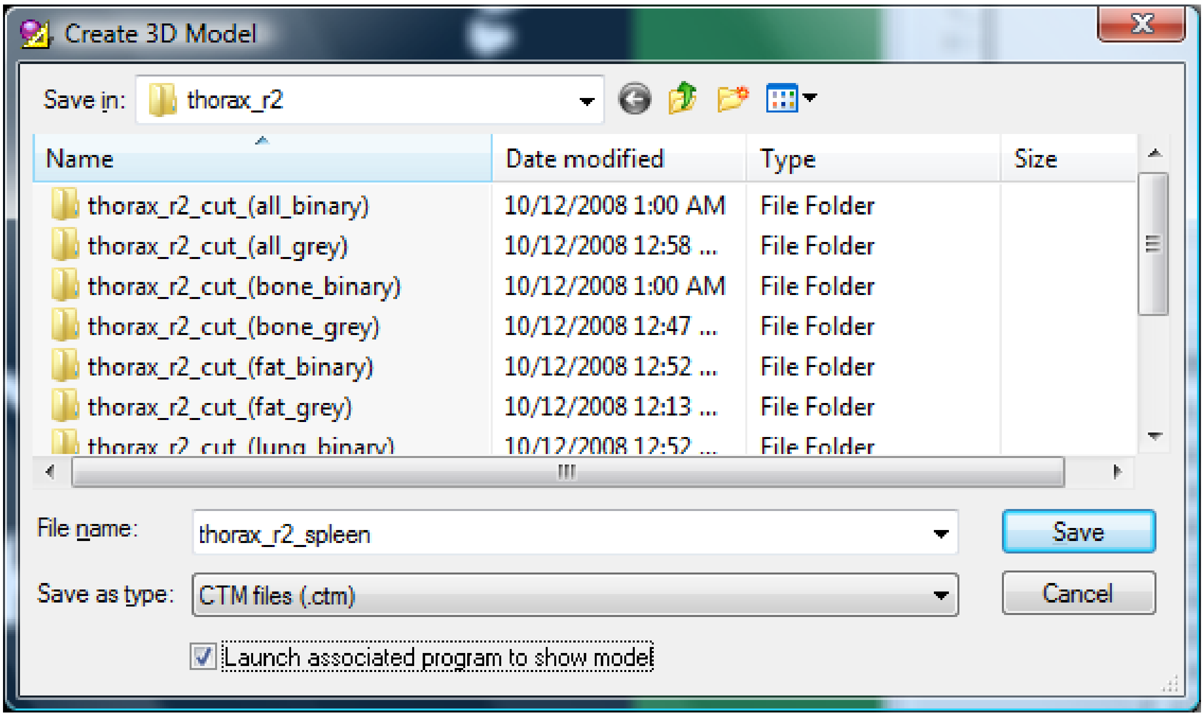
A note of caution: it is not always necessary to make a model of the whole of a scanned object: indeed this is sometimes challenging due to the very large file size that such a model would often comprise, in terms of surface facets and GB on disc. Inner structure is often better revealed by creating a model not of the whole object but of a slice through the object (using a simple rectangular ROI in CTAn) or some other appropriately chosen volume of interest ("VOI"). However if it is necessary to make a visual model of most or all of a scanned dataset, it might be pragmatic to resize the model downwards in resolution (upwards in voxel size) prior to model creation in CTAn; a smaller file size will make the model more responsive and easier to work with. At the time of writing, a model that is practical to work with means with a file size of about 1 GB or less.

Concerning computer hardware, it is strongly recommended to employ a PC with a 64 bit version of Windows, preferably Vista (and in future Windows 7). A minimum of 8 GB RAM is advisable for optimum performance. A good graphics card to use is one that can engage the PC's own memory, such as (at time of writing) the Nvidia NVS290.

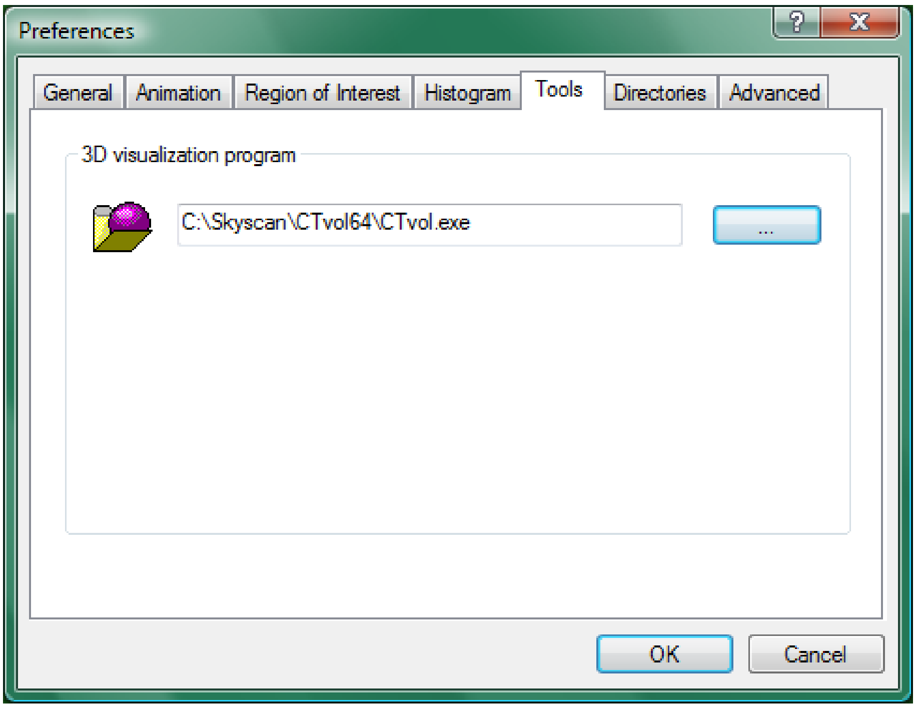
2. Starting CTVol

The program does not require special installation on the computer, it runs from the single *"CTVol.exe"* file which can be copied into any directory on the hard disk or it can start from a network drive. It can be made into a desktop icon (right-click on *ctvol.exe* and select "send to desktop (shortcut)". The first start-up might need to be under a user profile with administrator rights. Subsequent start-ups have no such restrictions and can be carried out from profiles with limited rights.

Please note that CTVol can be started directly from CTAn. In the dialog box for saving a 3d model file (at the 3rd, binary page of CTAn) there is a tick box at the bottom. If this is selected, then after completion of saving of the 3d model by CTAn, that model is then opened automatically in CTVol.



For this “launch associated program to show model” function to be available, there is an action required in the CTAn program, to set a link to the CTVol program in the CTAn preferences page. In CTAn, go to the file menu and preferences, and to the tab “tools”. In this tab, under the heading “3D visualisation program”, set the path to the executable file for CTVol (*ctvol.exe*) by clicking on the “…” button on the right (see the image below).



3. Creating 3d models

Surface-rendered 3d models for CTVol are created in the CTAn program. Please refer to the manual and other documentation for CTAn for details on model building. You have a choice of three surface rendering algorithms, which are described below; all three are based on binarised or segmented images within the volume of interest. Further, for models of all three algorithms, three file formats are available for saving the models, two of which are proprietary SkyScan model formats (CTM and P3G) and one is the industry standard stereolithography (STL) format. Model filenames have extensions corresponding to these formats.

3.1. Rendered model format

When building a model in CTAn, there are three algorithms: Marching Cubes 33, Double-Time Cubes or Adaptive Rendering.

Marching Cubes is a surface building algorithm based on an explicit hexahedral voxel model developed by Lorensen and Cline (1987).

Double Time Cubes is a Marching Cubes type method but with approximately half the number of facet triangles, with more smoothed surface detail (Bouvier 2000).

Adaptive Rendering is a sub-pixel smooth rendering method also employed in the SkyScan 3D-creator ("Ant") software.

Note that Marching Cubes 33 is the algorithm used by morphometric 3d analysis in CTAn. You can choose the algorithm in the dialog File / Preferences on the tab Advanced and the field 3D surface construction algorithm. If you choose Adaptive Rendering, configuration boxes will appear in which you should specify the "locality" and "tolerance" variable parameters.

The Locality and tolerance parameters for adaptive rendered 3D models

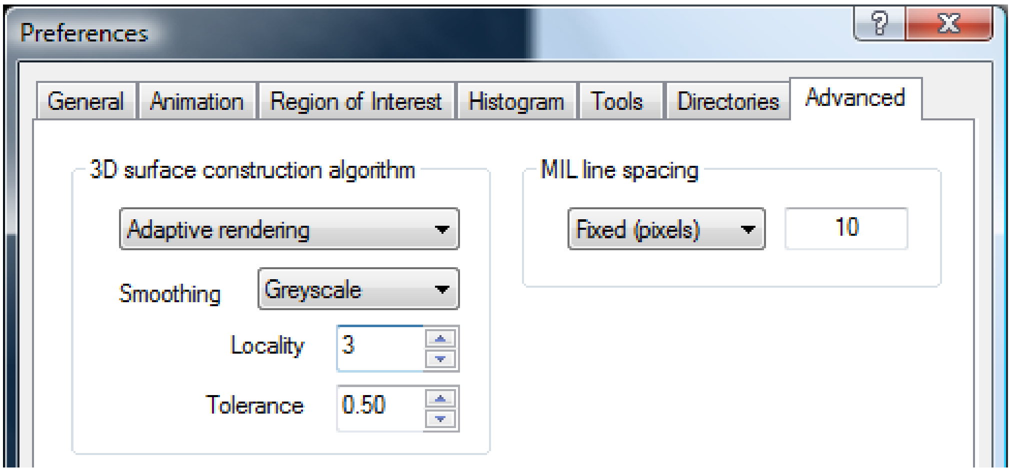
Locality: this parameter defines the distance in pixels to the neighboring point used for finding the object border. Increasing this value allows "jumping" through noise on the object border, but small objects (often created by binarisation of noise) with size less than this parameter will be lost.

Tolerance: this parameter defines the sub pixel accuracy with which the object border is delineated. Reducing this number makes the model more smooth and accurate, but increases the model file size (and the time taken to make the file).

Smoothing

The item "smoothing" also appears in the preferences advanced tab when the adaptive rendering algorithm is selected. Smoothing here really means "smooth interpolation", referring to sub-pixel surface interpolation in the case of the adaptive rendering algorithm, and there is a choice of two methods of this smooth interpolation. The two choices available are "binary" and "grayscale". "Binary" means that the model takes as input the binarised (black and white) images, but resamples

(and oversamples) the surface pixels into “pseudo” halftone (grayscale) values, then interpolates between these pseudo grayscale pixel values. The “grayscale” option, by contrast, also applies sub-pixel interpolation but accesses the original grayscale pixel values at the model surface from the original reconstructed images, rather than binarised (and then resampled) surface pixels.



After “type of file” you can choose between the following file formats for the created 3D model file:

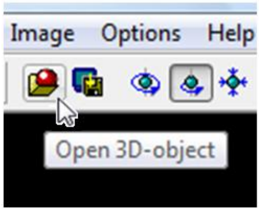
STL: Stereo lithography model format, a standard type accepted by much commercial software such as finite element analysis software (e.g. ABAQUS, ANSYS, Materialize)

P3G: A SkyScan model format. Where model size is small to medium this format is useful - the file size is relatively small and an additional smoothing option is available in CT-volume for p3g models.

CTM: A SkyScan model format. Model building in the CTM format is faster than in p3g, which can be useful for large models - such as from high resolution scans. However CTM models do not have the option for smoothing in CT-volume which is available for p3g models.

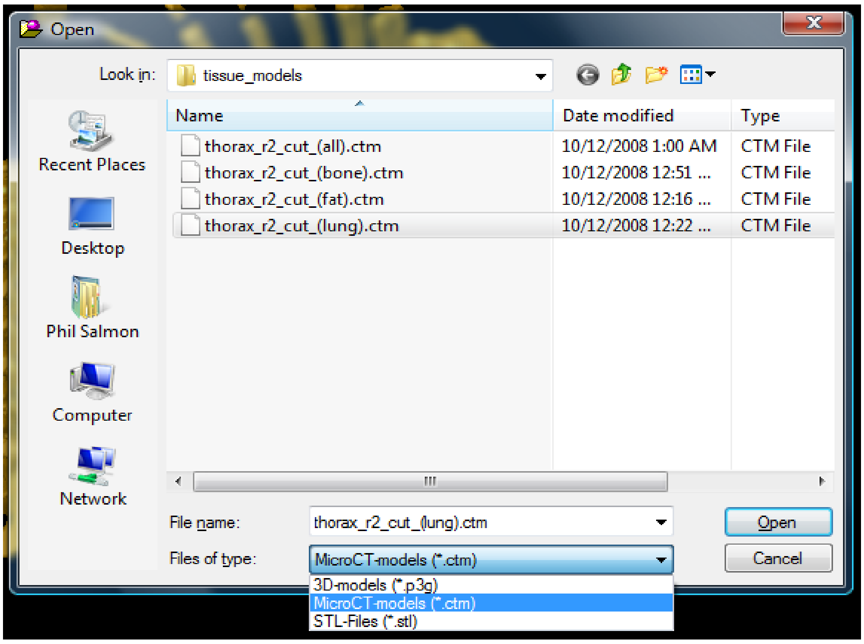
Please note that the model algorithm (Marching Cubes, Adaptive rendering etc.) and model format (P3G, CTM etc.) are not the same thing - this means that models made by any of the three algorithms can be saved in any of the three available formats. However models made by adaptive rendering should be saved in p3g format in order to visualise a smooth surface.

4. Loading a model into CTVol



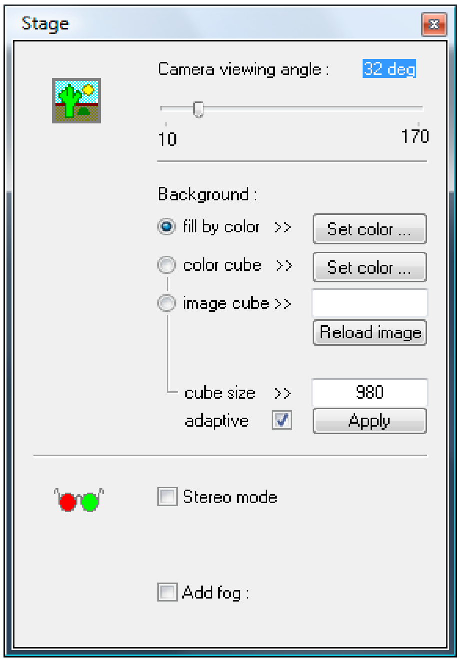
Click on the open button on the left of the button row. An open file dialog will appear. An image of this dialog is shown below. Please note: at the bottom after "files of type" there is a drop menu of the three model file formats, p3g, CTM and STL. The format selected is important, as only files of this format will be displayed - others will be invisible.

A progress bar will run in the bottom right of the CTVol window during loading. When



opening is complete the model will be displayed in the image.

Please note: it is recommended that in the stage properties dialog, activated by the button at the top: , you tick the box for "adaptive" under "cube size". The cube size is the dimensions of the 3d viewing volume within CTVol - having this sized appropriately to the loaded model avoids problems of the model not being visible on loading due to an inappropriate stage cube size.

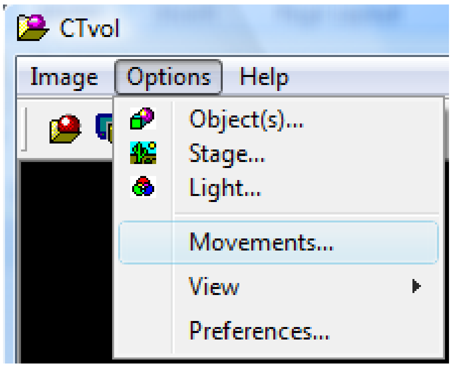


Once open, the model might need moving to a more appropriate 3d location and orientation. Note however that if you wish to open several models from the dataset, then for the models to retain their correct relative position, do not move any of the models until all the required models are loaded into CTVol. In the example shown below, models are shown for four biological tissue types of a mouse: lung, fat (adipose tissue), lean tissue and bone.

Movement and rotation of the model are described next.

5. Movement and rotation of a model

There are several alternative methods for model movement within CTVol.



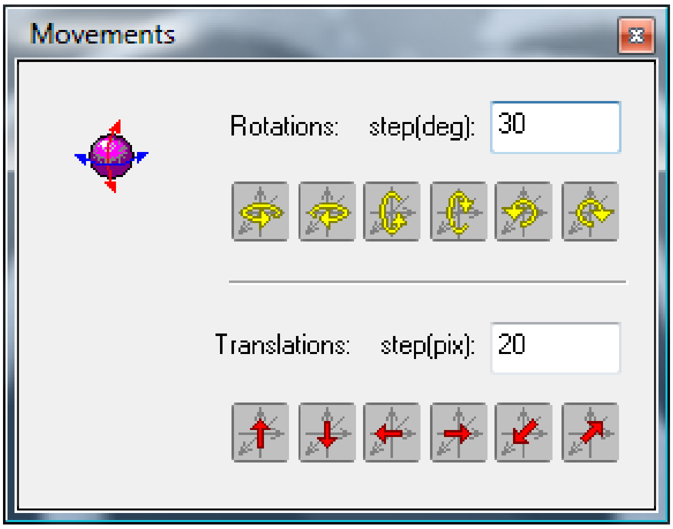
Mouse: drag-and-drop with the left mouse button controls translational movements up, down, left and right. Drag-and-drop with the right mouse button rotates the model. This rotation acts as a virtual trackball, with a cursor movement from left to right for instance making the front of the object model move left to right, and the back part move right to left.

The mouse wheel: the model is moved forward (toward the viewer) or back (away from the viewer) using the mouse wheel. The choice as to whether the model moves forward or back in response to - for instance - the forward roll of the mouse wheel, is set in the preferences window of CTVol.

Control dialog box: Under the options menu there is a “movements” item which opens a control window (shown below) which has buttons for all the translational and rotational movements. This window allows you to set exactly the number of voxels for each translation move and the degrees for each rotation move: this accuracy is useful when it comes to making animated movies.

Keyboard buttons:

Note that there are three rotational axes of rotation (in the XZ, YZ and XY planes). All these can be implemented using number pad commands, and two rotation axes using the keyboard arrows. The following keyboard buttons implement model translational movements and rotations:



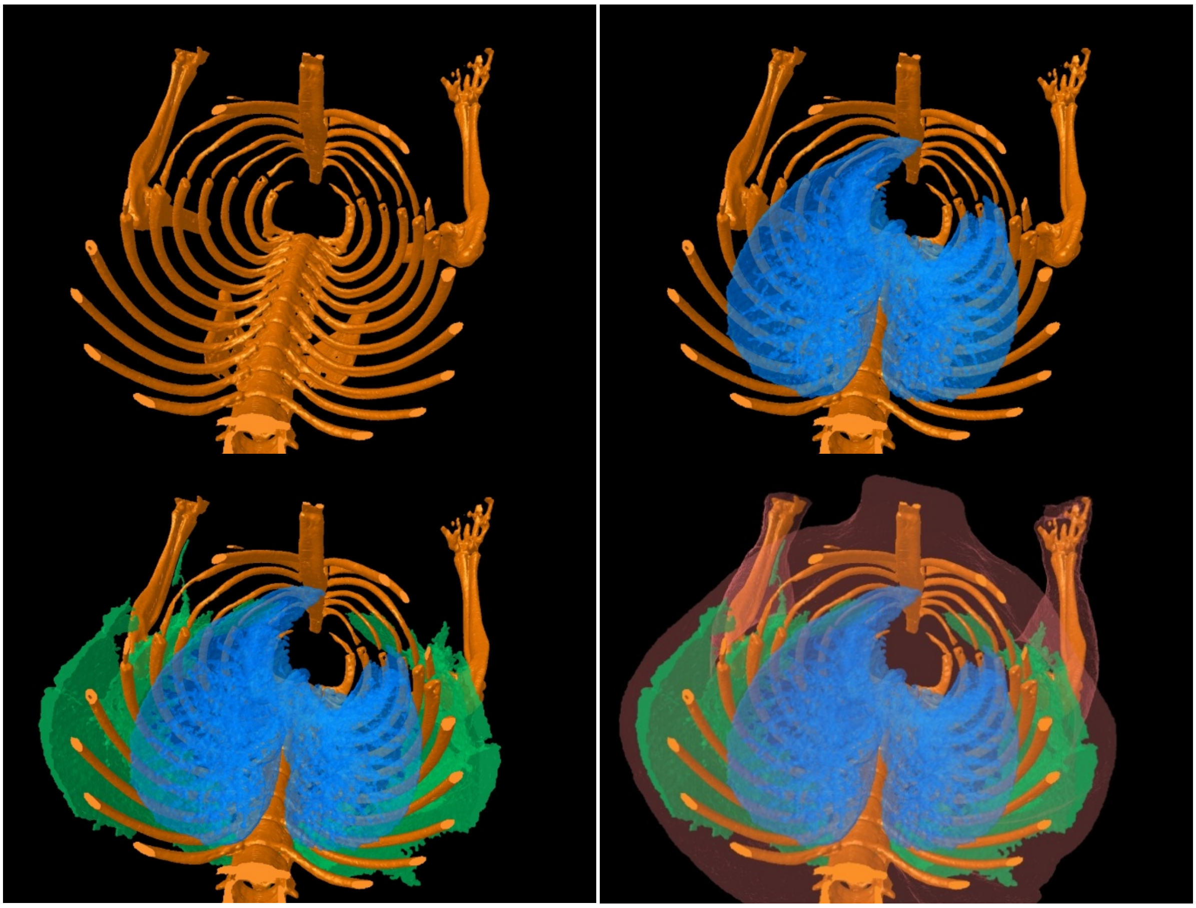
Keyboard arrows:

⇨ Right arrow: rotates model by moving the model front (close part) to the right (distant part or back moves to the left). XZ plane.

⇦ Left arrow: rotates model by moving the model front (close part) to the left (distant part or back moves to the right). XZ plane

⇧ Up arrow: rotates model by moving the model front (close part) upwards (distant part or back moves down). YZ plane.

⇩ Down arrow: rotates model by moving the model front (close part) down (distant part or back moves up). YZ plane.



When opening multiple models from the same dataset, such as these tissue models from an in vivo mouse scan (bone, lung, fat, all soft tissue), do not move the models until all models are loaded; after this, models can be moves as required, either separately or together.

Number pad key commands:

Number pad keys 4 and 6: horizontal rotation in XZ plane.

Number pad keys 2 and 8: vertical rotation in YZ plane.

Number pad keys 7 and 9: Clockwise (9) and anticlockwise (7) rotation in XY plane.

Translational movements:

U: The keyboard letter “U” moves the model up;

D: The keyboard letter “D” moves the model down;

L: The keyboard letter “L” moves the model to the left;

R: The keyboard letter “R” moves the model to the right;

F: The keyboard letter “F” moves the model forward (closer to the viewer);

B: The keyboard letter “B” moves the model back (further from the viewer).

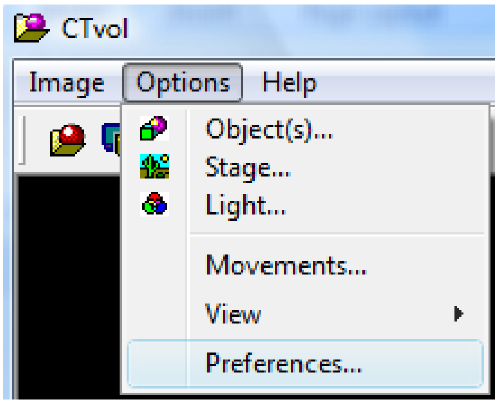
Please note that these keyboard letter commands are not case sensitive.

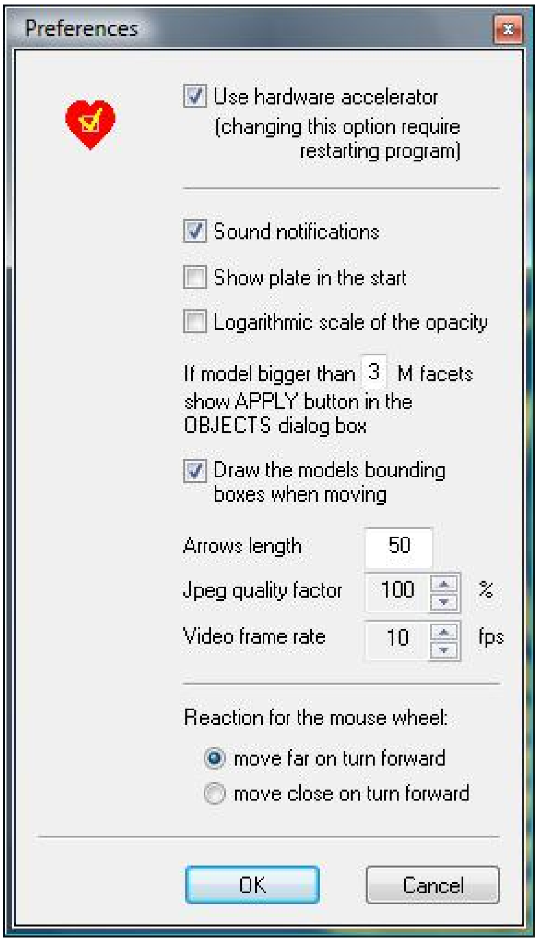
The degree of each rotation command and the distance (voxels) of each translation command is set by the user in the movement dialog box as shown above.

Please note also that the number pad is not present on a laptop PC.

6. CTVol preferences

Before moving on further in the operation of CTVol, a note on the preferences window and its contents. Preferences are launched under the options menu.





“**Use hardware accelerator**”. This engages the PC’s graphics card. This can speed up operations of CTVol. However occasionally some graphics cards can cause problems with the operation of CTVol, in which case this option should be deselected.

Please note: it is important to check that the correct and most up to date driver software for your graphics card is installed. This can make a difference to the operation of CTVol with acceleration. Go to the web site of the vendor of your graphics card (e.g. NVidia, ATI) and find the driver for your model of card[1].

“**Sound notifications**”. Turn on or off sound effects accompanying actions.

“**Show plate in the start**”. A plate model is always generated by CTVol together with your loaded model. The purpose of this is to perform cutting of the model. You can

[1] Please note: when the hardware accelerator is selected, this may affect how screen images of models are saved. Any window or dialog box open in front of the image pane of CTVol may be included in any saved images. This should not occur in the latest version of CTVol.

choose whether the plate model is visible or not. By default it is invisible. To make it visible, check the “visible” box in the object dialog .

“**Logarithmic scale of the opacity**”. This option makes the slider in the object properties which controls opacity – and its corollary, transparency – work in a logarithmic rather than linear way. This reflects the nature of the visible perception of transparency. A model with only 50% opacity looks not much different to 100% transparency, only below about 20% does it begin to really look transparent and reveal enclosed models.

“**If model is bigger … show APPLY button**”. This option is helpful for large models. When model size is large, it can take some time to respond to changes in the model properties, such as colour, texture and transparency. This “apply” option means that, when it is activated, a change made by the user in any of the model properties will not be implemented until the “apply” button is then pressed. This means that you can make several changes, then press “apply” and all the changes will be implemented together. This will be quicker than waiting for the program to implement each of the changes individually.

In the box after “bigger than” and before “M”, enter the number of million facets that will be the threshold; models with more than this number of facets will have the “apply” button activated.

“**Draw the model’s bounding boxes when moving**”. If this is selected, then when you apply a movement to a model using the mouse, either translation (left mouse button) or rotation (right button), then transiently during the movement the bounding cube is displayed in place of the model.

“**Arrows length**”. The length of the XYZ arrows displayed for the plate when the corresponding option in the model properties window is checked.

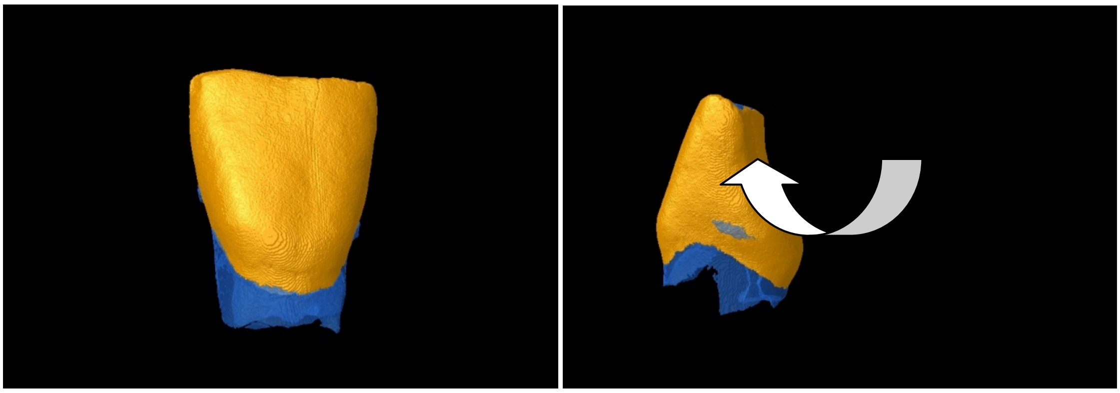
“**Jpeg quality factor**”. Saving of image files, either as single images or from the movie maker (flight recorder) can be in bmp or jpg format. If jpg is selected, a quality factor should be set corresponding to compression of the image of varying degrees for values less than 100%.

“**Video frame rate**”. The speed at which animated images in the flight recorder are saved. This determines the play-back speed of the avi movie file.

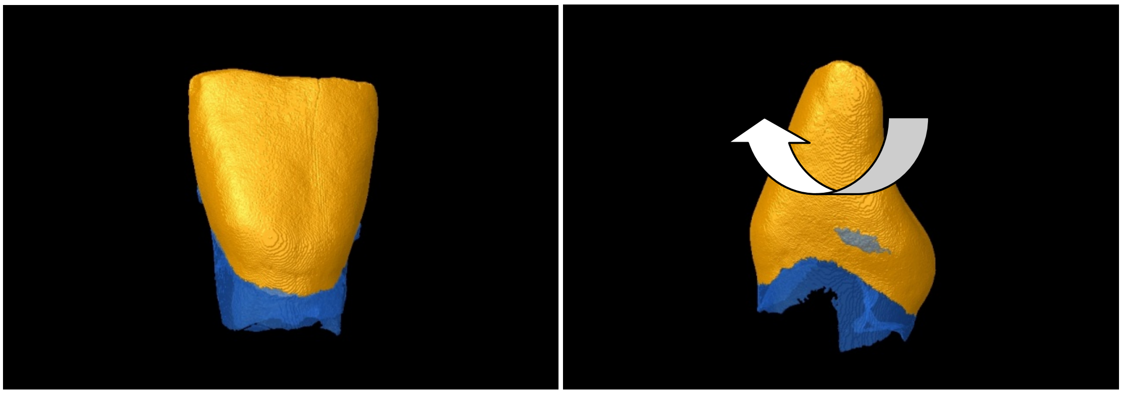
“**Reaction to the mouse wheel**”. Set the directionality of the mouse wheel controlled movement of the model forward and back. The forward roll of the wheel can either move the model further away (the option by default) or closer to the viewer.

7. Centering the model or models

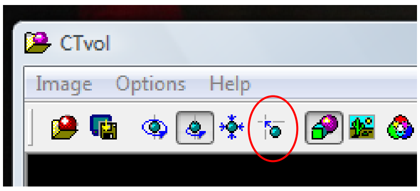
When a model is opened, the centre of rotation is taken as the centre of the dataset or volume of interest from which the model was created. Usually this is different from the central voxel, or "centroid", of the model itself. Rotation movements therefore result in the model orbiting around a distant point – as shown in the upper of the pairs of images of the tooth model.



Non-centered rotation (object rotates around distant point)



Centered rotation (object rotates around its own center).

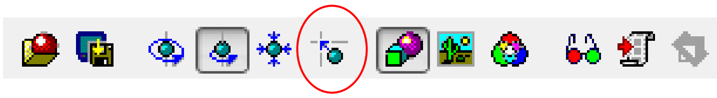


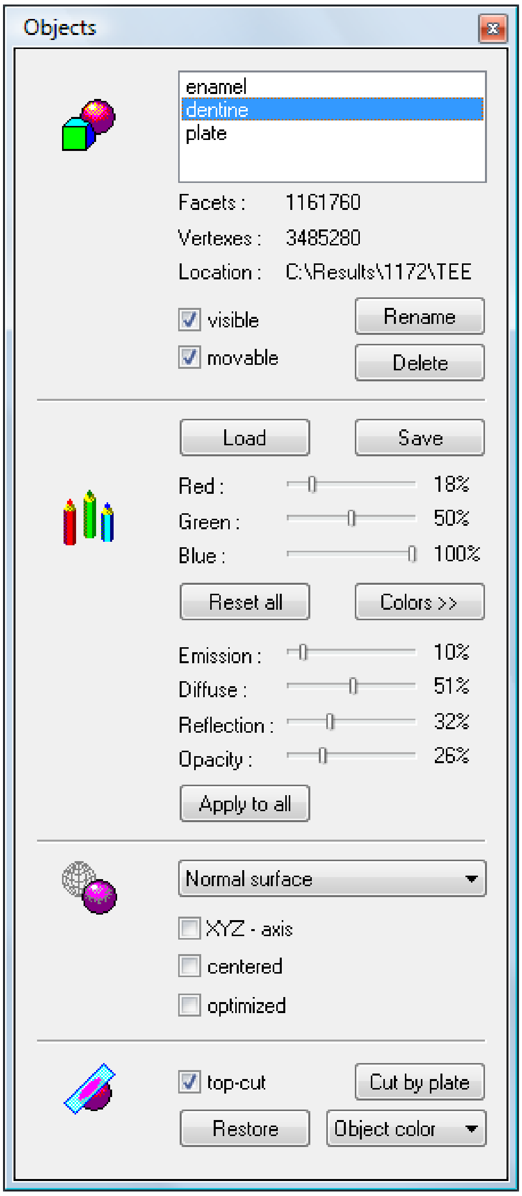
To achieve symmetrical rotation centered on the object's own centroid, you need to select the model in the model properties window, then click on the "move to object centre" button (right, circled). Then rotation of the model will be symmetrically around the object's own centre, as shown in the lower pair of images of the tooth model.

Please note – when more than one model is open, clicking on the "move to object center" button will center all open models to the centroid of the model which is selected in the object properties window (see next section). This is important since it allows centering of multiple models, for instance for the purpose of movie creation.

8. The object properties window

The object properties window is opened from the button as shown (red circle):





At the top of the object properties window is a list of the currently open models - “enamel”, “dentine” and “plate” in the example shown. Note that, at any time, only one model, shown highlighted in blue, is active (i.e. only that model will respond to applied changes to model properties). Below the pane listing open models, the number of facets and the number of vertices are listed for the selected model, as well as its file location.

Two tick boxes are present: “visible” and “moveable”. By default, both are checked - with the exception of the plate model which, depending on preference selection, can be invisible by default. Unchecking “visible” makes the selected model invisible (this is not the same as changing transparency - see later), and unchecking “movable” renders the model unresponsive to movement commands.

The “movable” option allows multiple open models to be moved relative to each other, by making only a selected model or models movable, and others non-movable.

The “visible” option can be useful in reminding the user of the identity of the open models.

“Rename” and “delete” buttons apply these actions on the selected model.

In the model appearance pane, at the top are the three colour sliders corresponding to the three colour sensitive retinal cone cells in the eyes of most mammalian users of CT-volume, namely red, green and blue. Lower down are a set of four sliders controlling model surface properties, namely “emission”, “diffuse”, “reflection” and “opacity”. At the top of this part of the object properties window are two buttons, “load” and “save”. These allow the saving, and subsequent reloading, of the entire set of currently selected visual properties (colour, texture) under a chosen filename. The saved “material” property file is given the extension “.mat” .

Below the three colour sliders, are two buttons, “reset all” and “colors”. “Reset all” will return the all colour and texture parameters to their initial default values (boring grey colour). The “colors” button opens a window for choosing a custom colour from a colour palette.

SKYSCAN

There are four texture controls which will be described in turn:

Emission. This controls the level of light emitted from the volume of the object itself. This can be useful when objects are transparent (opacity <100%) but is probably best set to a low value when opacity is 100%.

Diffuse. This controls the brightness of the model on the basis of diffuse reflection. After colour adjustment, the diffuse light level often needs to be adjusted to attain an appropriate level of model brightness.

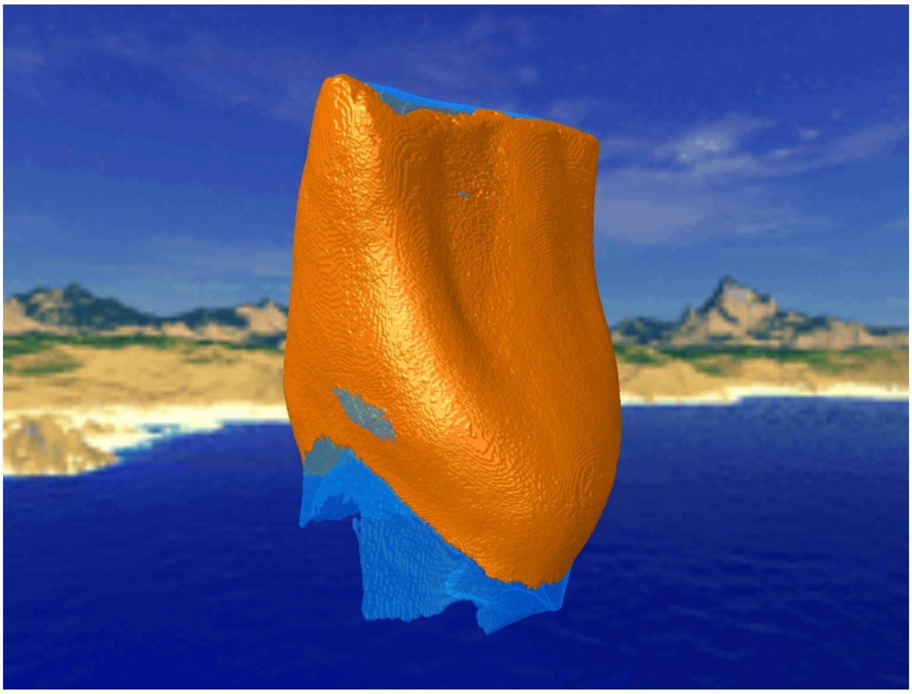
Reflection. This controls the shininess of the model, allowing adjustment between a matt (low reflection) and gloss (high reflection) texture.

Opacity. This determines the degree of opacity or its inverse, transparency, of the selected model.

The Apply buttons

Under the texture control tabs, there is a button "apply to all". This applies the currently selected colour and texture parameters to all open models.

Sometimes a second button "apply" will appear. This is when the size of the open model in terms of number of facets, exceeds a threshold number which is set in the CTVol preferences (see preferences item, section 6 above). When this apply button appears, then moving one of the colour or texture sliders will not result in immediate change to a model. Instead, implementation of the change now requires pressing of the apply button. This gives the advantage that several changes can be made to the colour and texture, and these changes then implemented together. This is useful when, with big models, implementing parameter changes can take more time; the user does not need to wait for each change to be made separately, but they can be made together.



Surface rendering types and other options. In this part of the object properties window there is a drop menu of several types of model surface facet rendering. These are:

Wire model: This can sometimes be useful for showing surface detail at very high magnification;

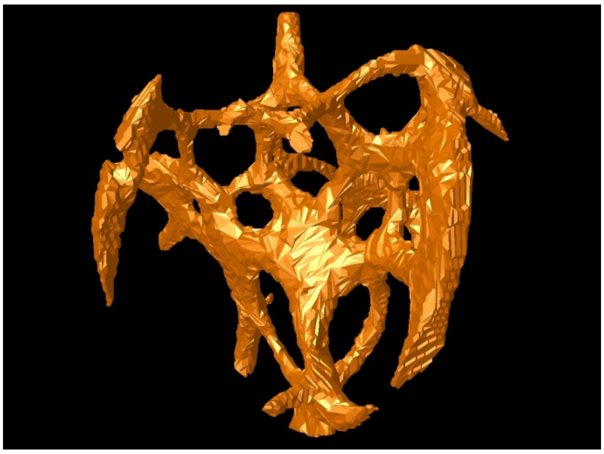
Triangular facets: This shows the boundary voxel facets explicitly as flat planes, giving a very sharp appearance;

Normal surface: In this rendering method, which is set as default, voxel surface boundaries are slightly rounded to give a smoother texture.

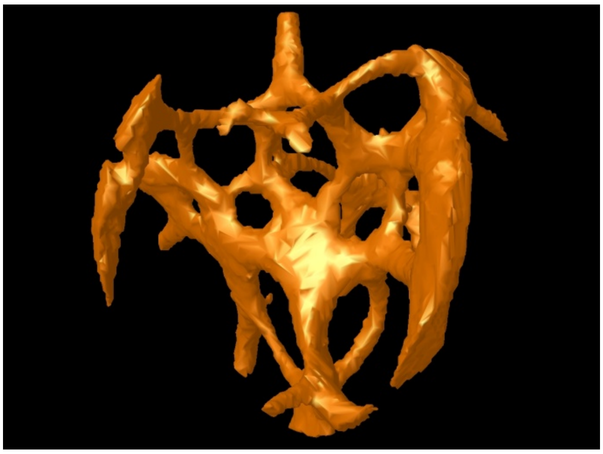
Smooth surface: This option is only available if the model is built in the p3g type. The voxel boundaries are further smoothed, giving a smoother rendering.

The four surface rendering types are illustrated below, in the example of a model of a small volume of trabecular bone.

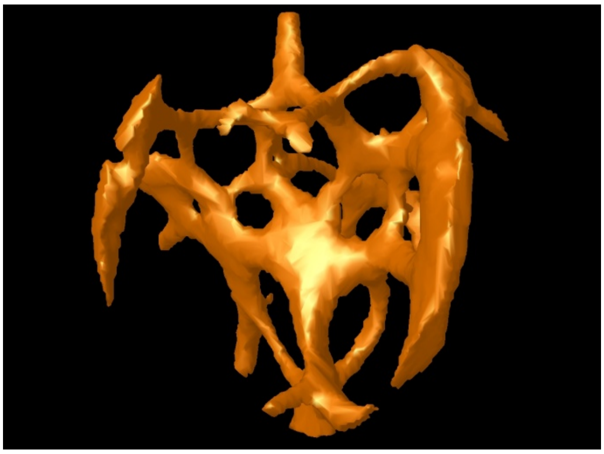
Wire model



Triangular facets



Normal surface



Smooth surface

Other options

XYZ axis: Checking this option makes a triple-arrow visible, showing the x, y, and z axes. The blue arrow shows the z axis and points in the direction that is considered as “up” by the program. This is necessary when cutting models using the plane, so

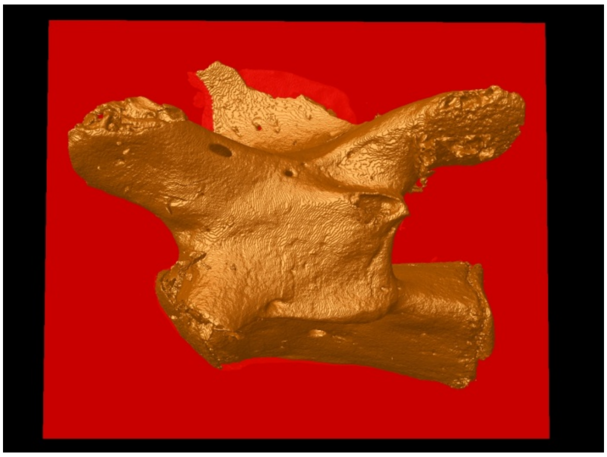
that the user knows which side is going to be cut (see later section). Occasionally the XYZ axis may appear inside the model volume and thus not be visible; in this case, you should reduce opacity of the model until the XYZ axis arrows are visible. If the arrows are small, move closer (by the mouse wheel operation).

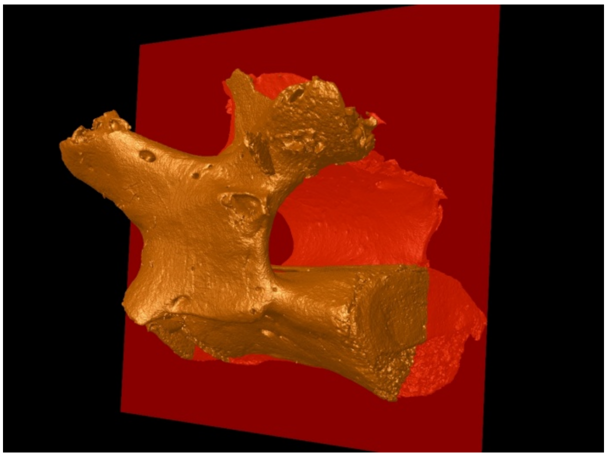
Centered: This button centers the selected model: after pressing "centered" you must then press "apply", Please note: this function centers the currently selected model only. The alternative button at the top button row of CTVol, the "move to object center" button (see section 7), centers all open models to the centre of the selected model. This latter, upper button is preferable when the user needs to center multiple models.

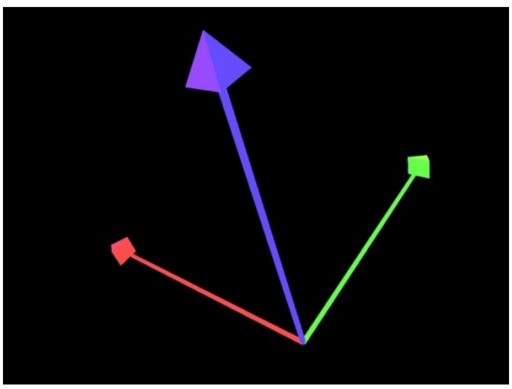
Optimised: It was mentioned above that the "smooth surface" option is available only for models saved in the p3g type. If a CTM type model is open, it can be converted to a rendering similar to the p3g type by clicking on "optimise" then the apply button. After this, the "smooth surface" option becomes available for the CTM model also. Note, however, that the optimise process can take quite a long time, depending on model size.

Cutting a model with the plane.

When you open a model in CTVol there is always a plate model also open, although it may be invisible (depending on the preference selection). The plate can be used to make a section cut through the model, by removing the part of the model on one or other side of the plate. How this is done is illustrated below in the example of making a cutaway model of a mouse vertebra.







Having made the plate visible and selected its properties (here red color and transparent) the next step is to position the plate relative to the vertebra model in the plane desired for the cut. This is done by alternately making the model moveable and non-moveable, and moving the plate to the necessary position.

With the plate correctly positioned, here for a sagittal cut, it is necessary to know which side of the plate will be removed by the cut. CTVol

defines one side as up, the other down. To find out which is which (it may not be obvious after repeated rotations and movements of the plate), first make all other models invisible (deselect the visible tick-box) and, having selected plate in the model list at the top, select the XYZ-axis option near the bottom of the object properties window.

Now if you zoom in on the plate you can see red, green and blue arrows indicating the X, Y and Z directions. The blue arrow shows the up direction.

Having determined the “up” side of the plate, make the model again visible. In this example of the vertebra there is a single model only. Where multiple models are open, each model needs to be cut separately. In this case we wish to cut away the part of the vertebra on the top side of the plate. Select each model, then go to the cut by plate item at the bottom of the object properties window. Tick the box for “top-cut” and click on “cut by plate”.

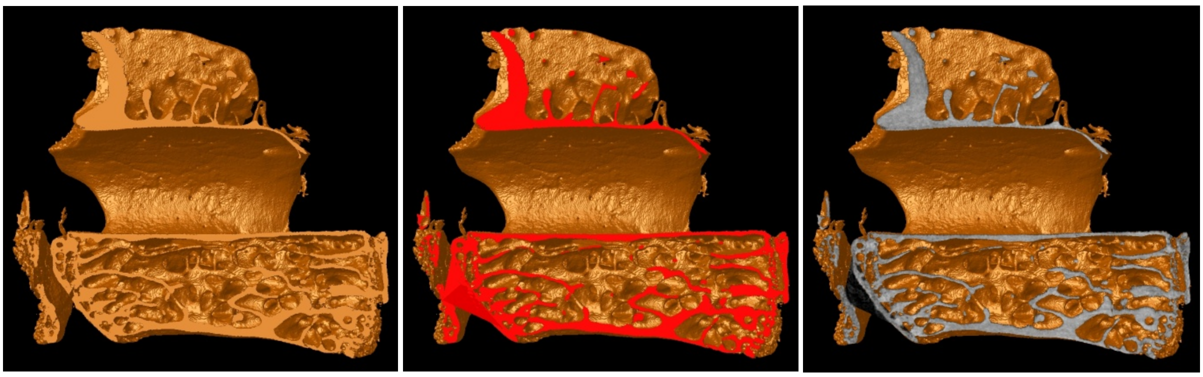
Note that the cut surface can be colored in three ways:

(1) the model color;

(2) the plate color

(3) the model “texture” this option means that the grayscale (“halftone”) voxel values from the original dataset are painted onto the cut surface for a realistic effect.

The results of painting the cut surface in these three ways are illustrated below:

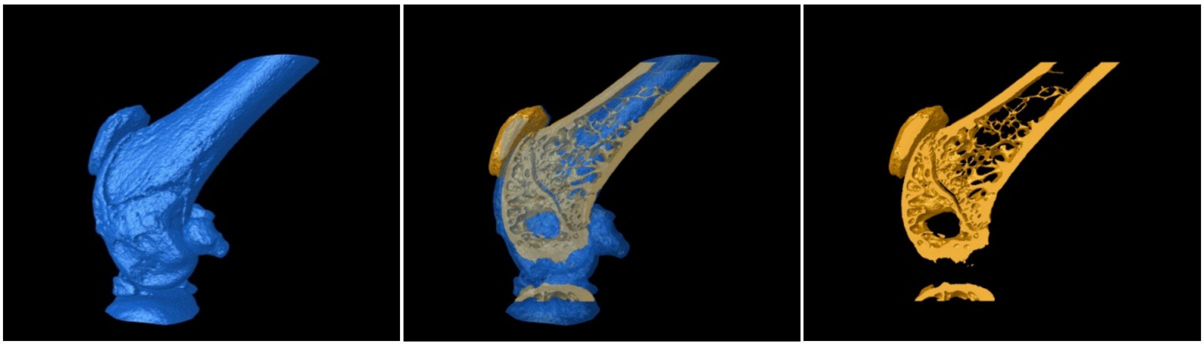


The tooth following cutting by the plate, with the exposed surface of the solid enamel colored by the model color (left) or the plate color (right).

A cut that has just been performed can be reversed by clicking on the “restore” button.

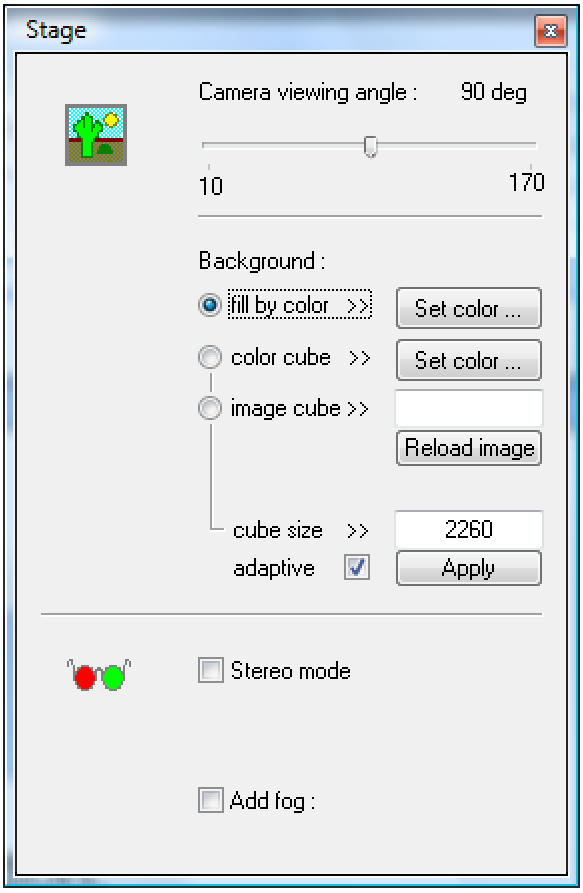
A further note: there exists an alternative to cutting open a visual model using this plate cutting tool. That is to create the model in CTAn using a volume of interest that cuts the model in the desired way. Sometimes for instance a simple rectangular volume of interest allows a slice model to be created which can be more revealing of inner detail than the model of the whole object. An example is shown below: the model of a mouse femur is shown (blue) and also the model created from a rectangular volume of interest within the femur dataset (gold).

SKYSCAN

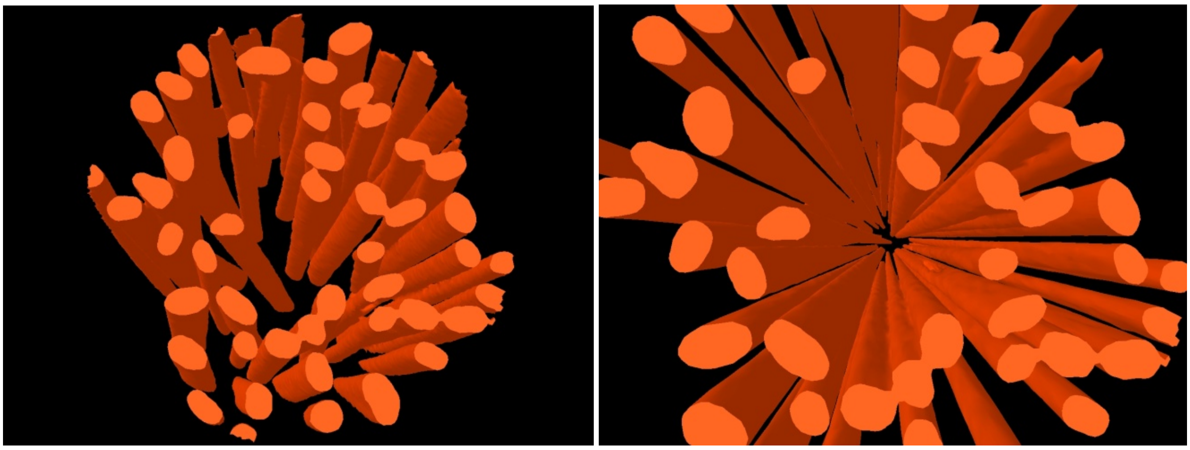


Alternatively, more complex non-planar cuts can be made in this way taking advantage of the volume of interest selection tools in CTAn.

9. The stage properties window

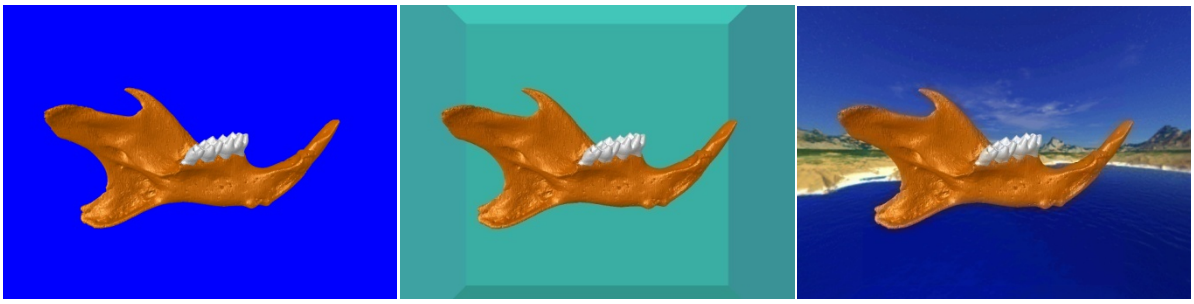


At the top of the stage properties window is a slider to determine the camera viewing angle. The effect of viewing angle is illustrated in the two images below. A wide viewing angle (large value in degrees) is equivalent to wide angle in photography - a wider scene is viewed. This is easier to see if the color cube or the image cube backgrounds (see below) are selected. Note that at high viewing angles, the object becomes distorted when close to the camera due to greater magnification closer to the viewer. Wide angles are appropriate for viewing multiple objects while narrow angles (e.g. 30-45 degrees) are more suitable for viewing single objects.



Camera viewing angle: A narrow angle (30 degrees, left) and a wide angle (130 degrees, right). High viewing angles stretch and distort the part of the object closest to the viewer.

Background. Three backgrounds are available. They are illustrated in images below:



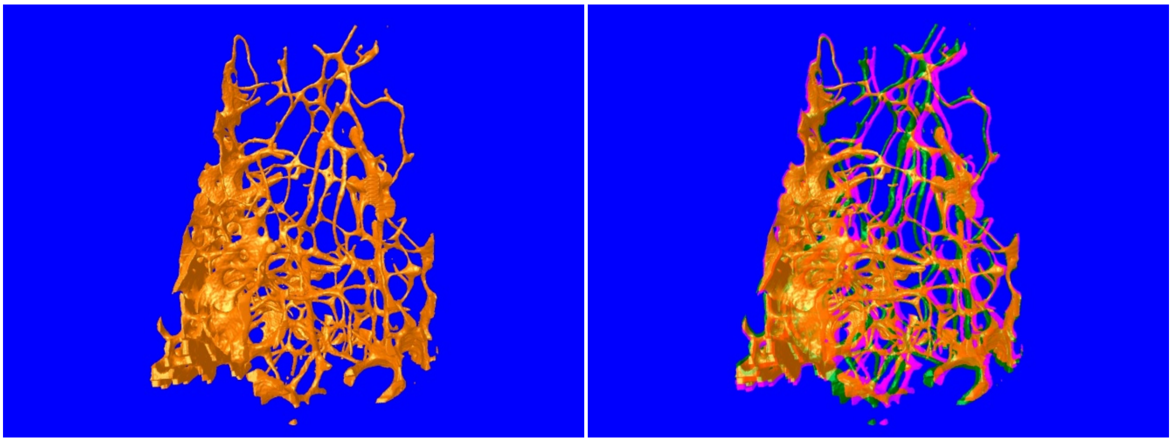
The three background scenery options: fill by color (left), color cube (centre) and image cube (right).

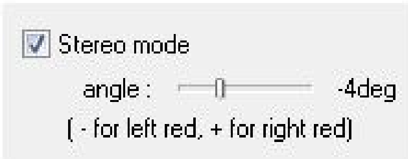
Fill by color: Here a single background color is set, by clicking on the “set color” button. This sets an infinite viewing space without boundaries.

Color cube: With this option the boundaries of the viewing cube are visible, as different shades of color. The dimensions of the cube are set as “cube size” (see below). Again the color can be chosen by the “set color” button.

Image cube: With this option an artificial scenery of sea, coastline and sky is provided. This scenery can help to provide a horizontal referent. Note that the user has the option of providing their own image cube. Six square bitmap images are required, corresponding to the six faces of a cube. They need to be named with 6 filenames with a prefix and ending in: “…front, …back, …left, …right, …top, …bottom” and placed in the directory with CTVol. For example “skyfront”, etc.. With these images in place, the user should write “sky” (or other prefix) in the box after “image cube”, and click on “reload image".

Cube size: This entry determines the size of the viewing cube. The cube should be sized appropriately for the model and the dataset from which the model was made. The easiest way to guarantee this is to select “adaptive”. Once this is selected, for models that are subsequently opened the cube size is set automatically on the basis of the size of the parent dataset.



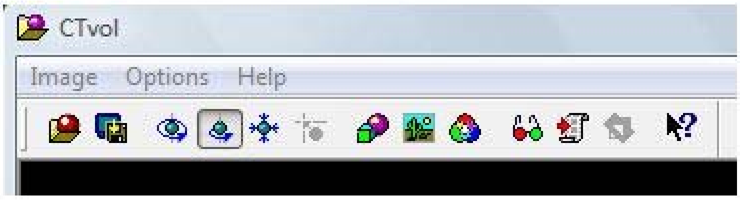


Stereo mode. Clicking on this option transforms the image of the model into the green-red shifted mode which, when viewed with red-green colored glasses, gives a 3D depth to the image. With this option selected, a dialog box opens allowing adjustment of the parallax angle between the right and left eyes (determined by distance of viewer from the screen; typical values are 2 - 4 degrees).

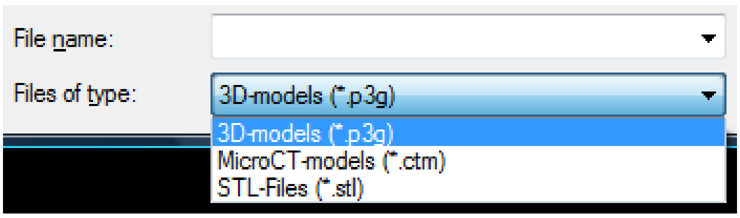
Add fog: Adding fog applied a grey shading to objects with increasing effect further into the distance: the effect is to give a sense of distance.

10. The toolbar items

The top toolbar items will be described in turn.



 Opens a model. A dialog box opens in which the model name and type need to be specified.



 Saves the current screen image to a specified filename, in either bmp or jpg format.

 This button sets the movement controls to camera mode. This means that the camera moves around the object with rotational movements.

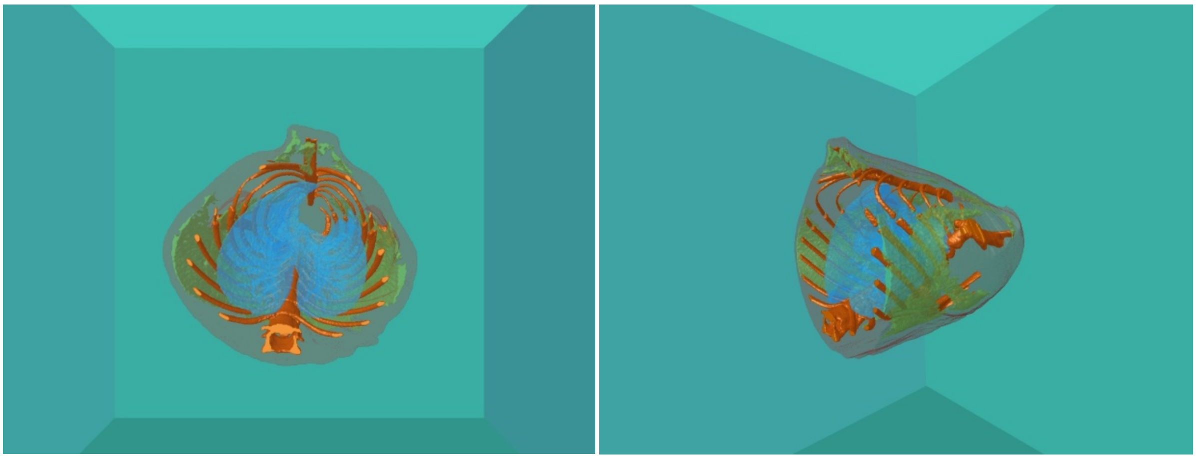
Note - with this mode selected, the left mouse button drag-and-drop action has a different effect: it changes the magnification of the model making it larger or smaller.

 This button sets the object movement mode - this is the default mode - in which rotation movements are applied to the object while the camera remains stationary. In this mode, left mouse button drag-and-drop movements cause translational movements of the model (not magnification changes as in the move-camera mode).

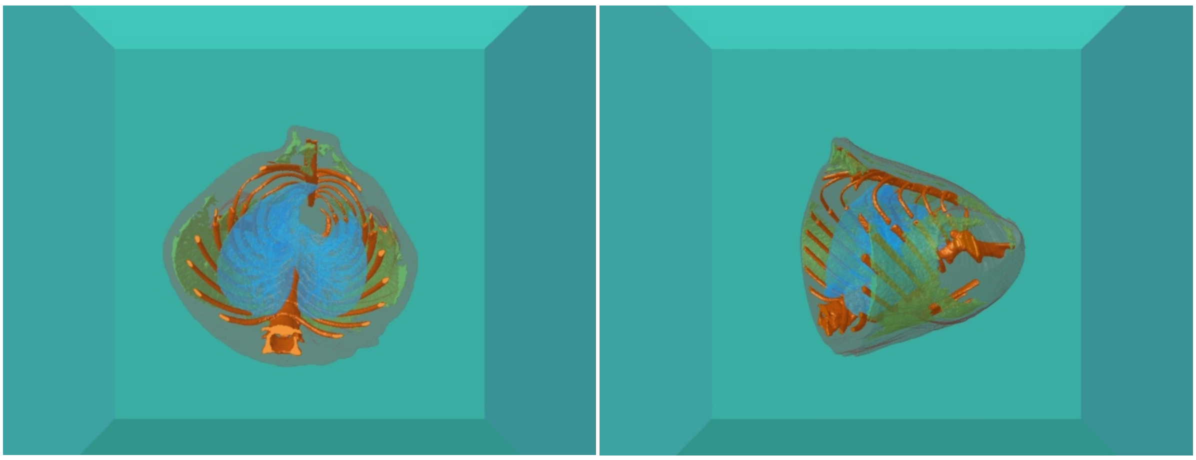
The effects of applying a right mouse drag-and-drop rotational movement in the “camera” and “object” modes, as just described, are shown below in the two pairs of images.

 Reset object position. This button resets the objects position to the initial position on loading.

 This is the “move to object center” button, an important function to centre all open models to the centre of the currently selected model; its use is described in section 7 above.



Models of a mouse tissues are rotated in the “move-camera” mode, from a starting position (left) to a rotated-left position (right).

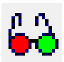


Models of a mouse tissues are rotated in the “move-object” mode, from a starting position (left) to a rotated-left position (right).

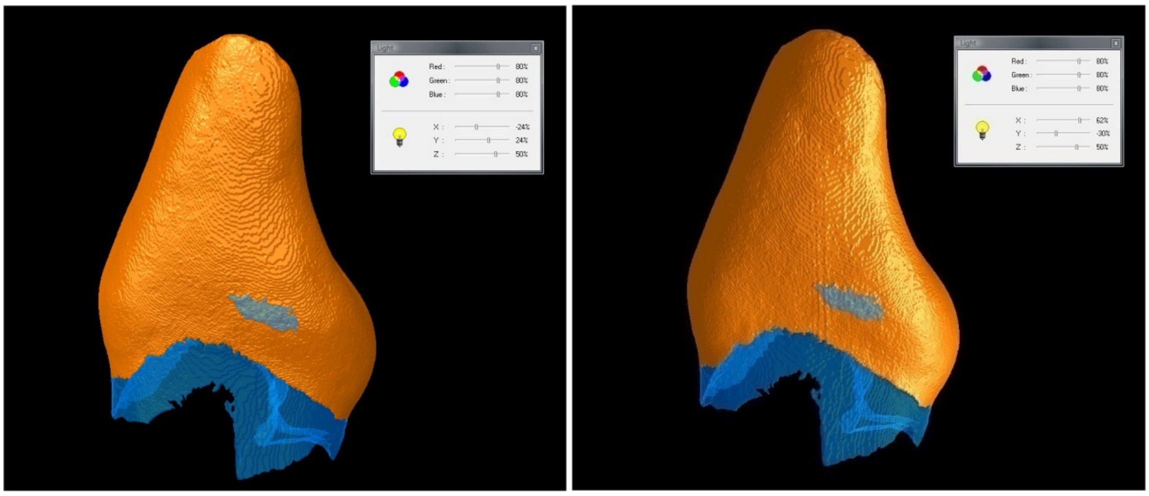
Note that in both object and camera viewing modes, movement of the mouse wheel moves the object forward and back in the same way.

The object properties and stage properties buttons (left and right respectively). These property dialog boxes are described in detail in sections 8 and 9.

The light properties button. This opens a dialog box that allows adjustment of the direction of lighting of the model. The effects of this adjustment are illustrated below. In the top of this dialog, the color balance of the lighting can be adjusted, and in the lower part, the direction of lighting can be changed.

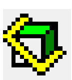


This button applies a red-green color shift to the image to allow 3D effect viewing with red-green (or red-blue) glasses. More distant objects have a larger red-green shift than near objects, giving a sense of distance and depth when viewed with the colored glasses.

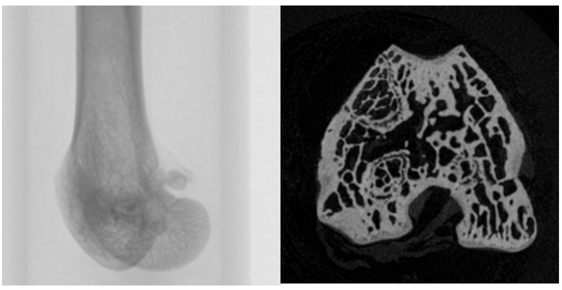


Changing the light direction: the default lighting (left) and lighting with the X and Y values reversed (right) so that the right side of the tooth model is preferentially illuminated.

Flight recorder. This is the function that allows creation of animated movies, and it is described in the next section (11).



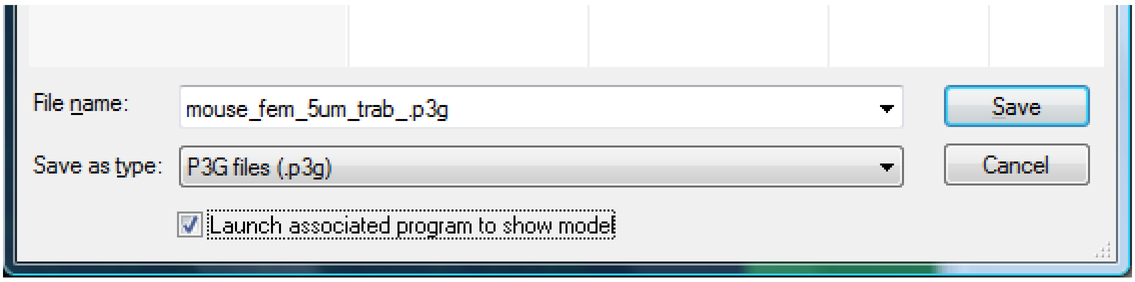
This button launches the function to reslice a dataset in the plane of the plate model, creating new reconstructed slices in the selected plane, or a maximum intensity projection (MIP) in the same plane. Please note: this operation is linked to the dataset



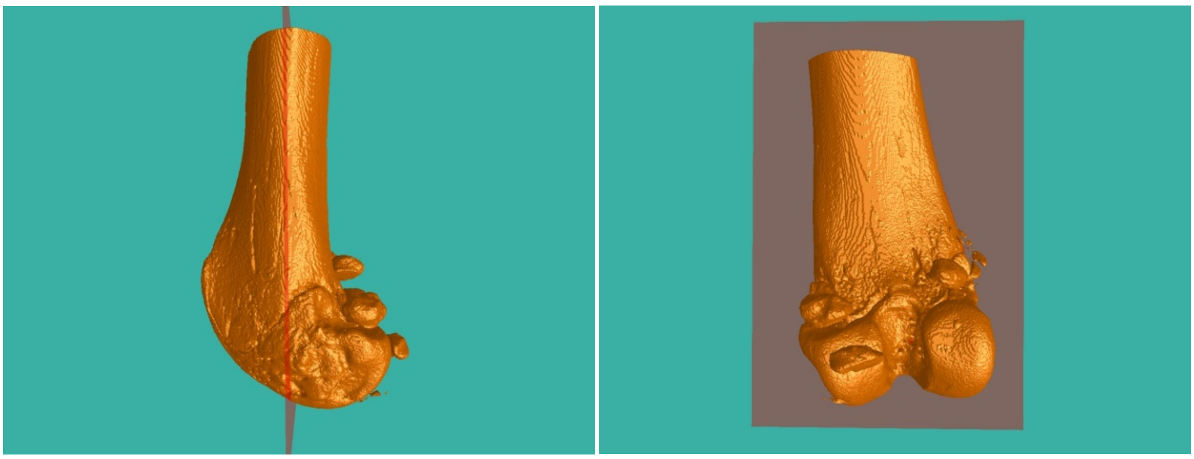
opened by CT-analyser and requires that CTVol is launched automatically at the end of model building in CTAn. An example is illustrated below, of the longitudinal (coronal) reslicing of the scan of a rat distal femur. An original crossection and the projection image from this scan are shown below.

Reconstructed crossection (left) and projection image (right) from a scan (SkyScan 1173) of a rat distal femur. The aim is to reslice this dataset in a selected coronal plane, using CTVol.

Referring to the CT-analyser program ("CTAn") where the 3D model is created, the dialog box shown in the image below (left) includes the tick box at the bottom, "launch associated program to show model".

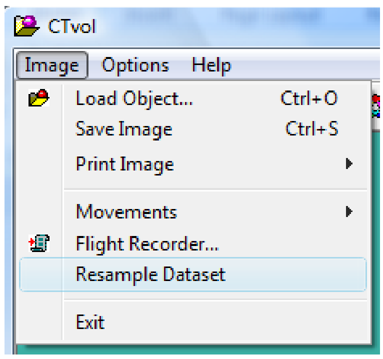


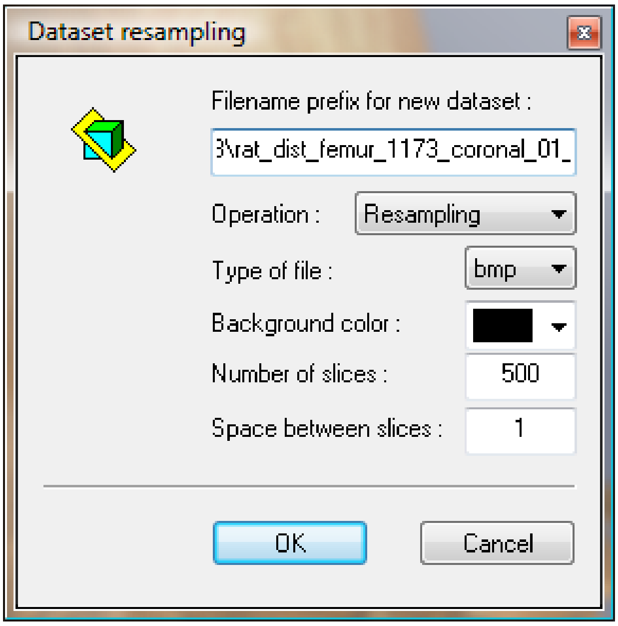
With the model created in this way, it then opens in CTVol with the reslicing function available - the button  is available. When models are not opened directly in this way then this button is grayed out.



With the model of the femur open, the plate can be positioned in the desired plane for a coronal reslice, as shown above. This is done by alternately making the model moveable and non-moveable relative to the plate, as shown in section 8 under the "cut by plate" part.

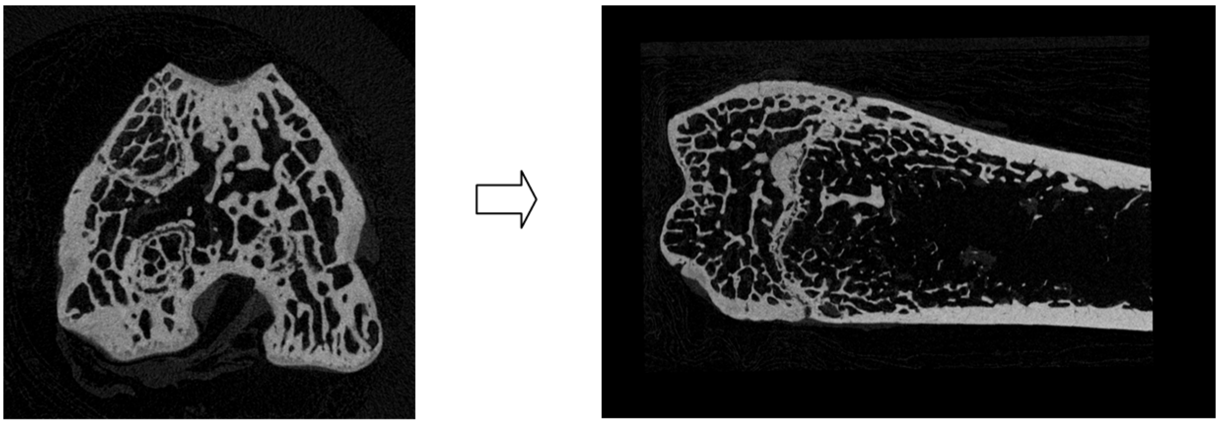
With the plate and model positioned correctly, go to the image menu and select "resample dataset" - or click on the corresponding button at the top bar. The dialog box shown below will open.



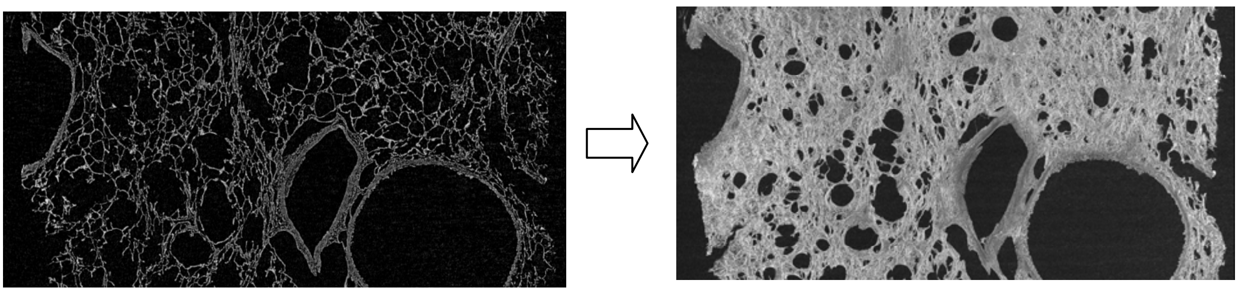


Fill in the required information including the file type, background color, number of slices and space between slices, as well as the required filename for the resliced images (here “coronal_01”). The reslicing operation will begin, its progress being indicated by a progress bar in the lower boundary of the CTVol program.

The resultant reslicing from a bone (rat femur) crossectional to coronal section is illustrated below. Also shown is the sampling of an MIP image from the reconstructed images of a lung tissue sample.



The reslicing of a bone (femur) dataset from the original transverse crossections (left) to coronally resliced sections (right) using the plate slicing tool in CTVol.



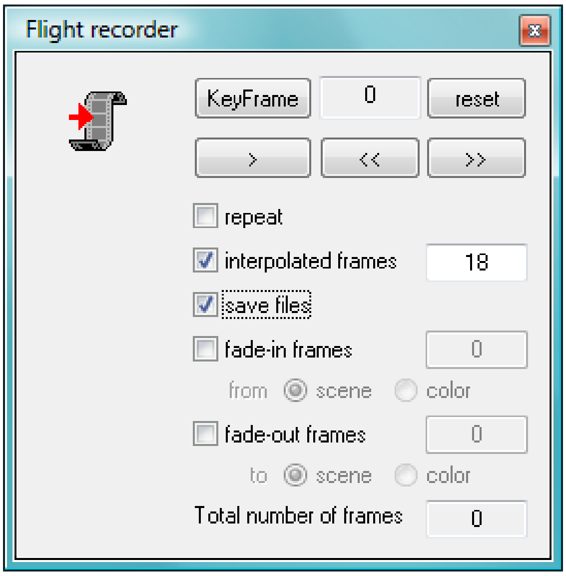
The sampling of a maximum intensity projection (MIP) of a lung tissue micro-CT dataset from the original transverse crossections (left) to the MIP image (right) using the plate slicing tool in CTVol.

Note that when both reslicing and making MIP images, the user should choose the appropriate number of slices to achieve the desired effect; sometimes the whole dataset can be resampled, at other times it is more appropriate to sample a limited number of slices.

11. Making movies – the flight recorder

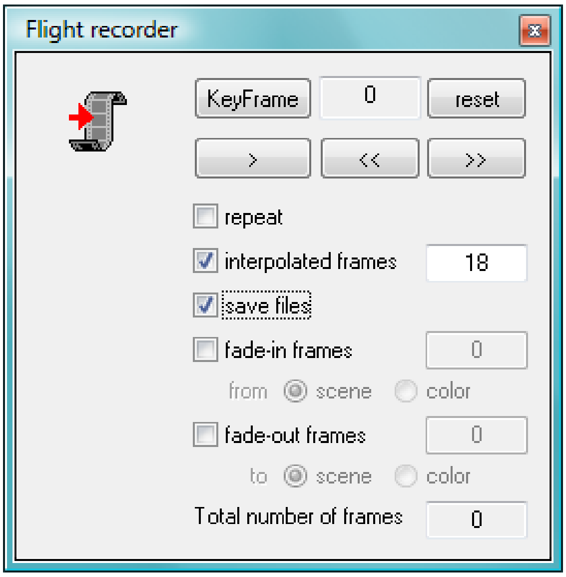
The movie making function is initiated by the "flight recorder" button. 

This opens a dialog box as shown below. The principle of the flight recorder is to set certain reference frames for the model(s) and its properties, called "key frames", by clicking the "KeyFrame" button after setting a number of consecutive scenes on the screen. The flight recorder will interpolate intermediate frames in between the key frames, with graduated changes to the model properties and spatial position between the two key frames.



After "interpolated frames" you should enter the number of frames that you wish to be interpolated between each pair of successive key frames. When you enter a value, the calculated total number of frames will appear at the bottom of the window, depending on the current number of key frames.

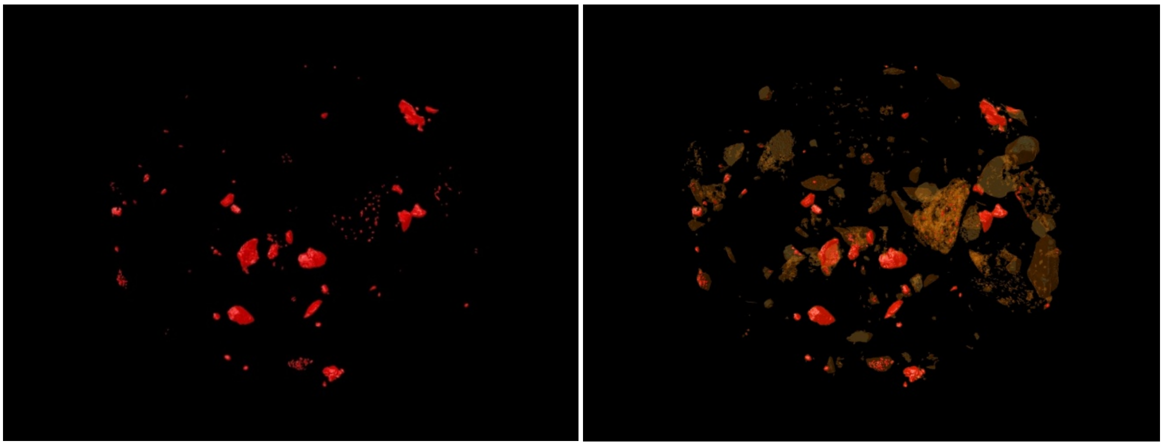
The creating of a movie from the scan of a volume of sand will be shown below. The sand was shown by the micro-CT scan (SkyScan 1172) to contain several density phases. Low, medium and high density phases were made into three models in CTAn. These were given different colors. The key frames that were set to make this movie are illustrated below (next page).



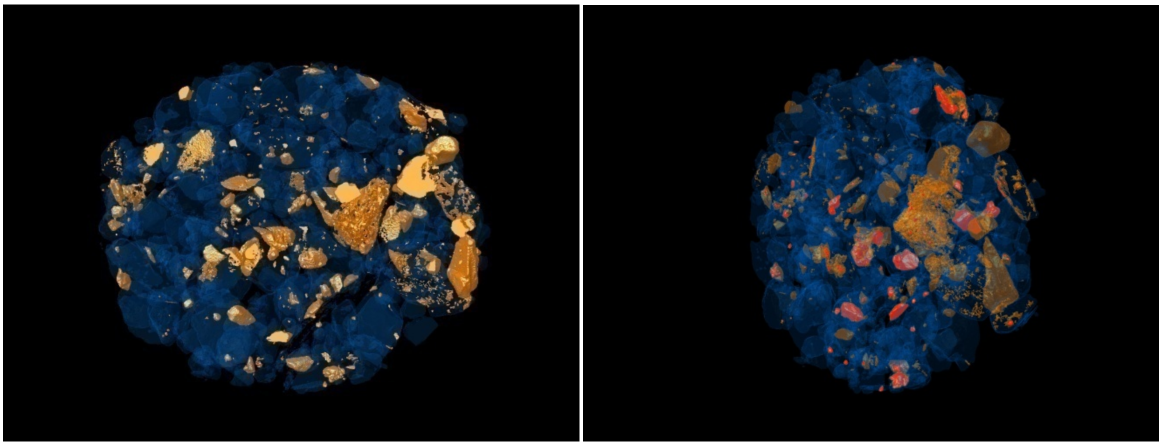
On completion of the setting of the key frames, the "save files" box should be ticked if you want to save the movie. Otherwise interpolated movie will be shown only on the screen. Then click on the forward play arrow button , and a save files dialog box will open. Specify a folder and filename for the movie.

Note that you will be given a choice of saving the model as a series of images in bmp or jpg format, or of saving the movie in avi format. In the case avi is selected, you will also have the option to choose a compression codec for reducing the avi file size: the available options will depend on what codecs are present on your computer. Codecs that are usually present and which work reliably are "Microsoft Video 1" and "Intel ITUV codec". With Microsoft Video 1 and some other codecs, there is a slider to select image quality: it is best to set this to 80 - 100%.

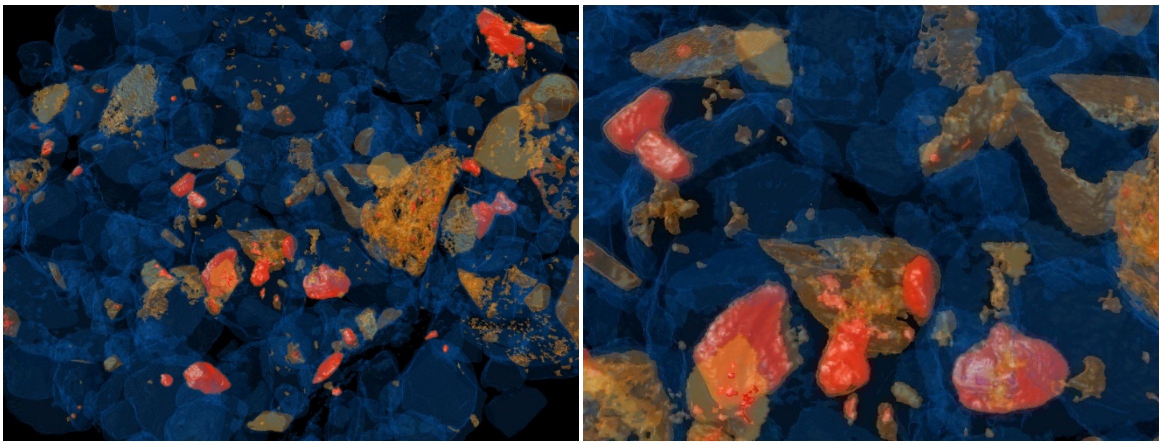
SKYSCAN



Key frame 1 (left) – the high density model is made opaque and red in color; Key frame 2 (right): a second, medium density model (yellow) is increased from zero to partial opacity.



Key frame 3 (left) – the medium density model is made fully opaque and the third, low density model (blue) becomes visible and transparent; Key frames 4-11 (right): with transparency set so that all three density models are visible, a series of rotations by 45 degrees are entered as key frames, to achieve a full 360 degree rotation.



Key frames 12 - 14: here the model is moved successively closer, so that the interpolated effect will be a “fly through”. One can select a hole between the grains and fly through it, navigating by the translation and rotation controls using either move-object or move-camera as appropriate.

SKYSCAN

Note that when rotating models for a movie, it can be helpful to use the tool for fixed degree rotations, to create smooth rotation, rather than the right mouse drag-and-drop rotation.

Fade-in frames:

You can choose to make the first key frame image fade in from total transparency or from a specific color at the start of the movie, by ticking this box. The fade-in will take place over the number of frames specified in the associated box.

Fade-out frames:

You can choose to make the final key frame image fade out to total transparency or to a specific color at the end of the movie, by ticking this box. The fade-out will take place over the number of frames specified in the associated box.

12. References

Dennis J Bouvier, *Double Time Cubes: a fast surface construction algorithm for volume visualisation.* Unpublished report, University of Arkansas, 313 Engineering Hall, Fayetteville, AR 72701, USA, 2000.

Lorensen WE, Cline HE (1987) *Marching cubes: a high resolution 3d surface construction algorithm*. Computer graphics 21 (4): 163-169.